to be presented at IGTC 2015, Tokyo, November 11 – 15 2015

A Summary of Computations of Ingestion at the University of Bath

Mike Wilson and Gary Lock

Department of Mechanical Engineering, University of Bath
Bath BA2 7AY, UK

ABSTRACT

Rim seals in gas turbines are used to reduce the potentially damaging ingestion of hot gas into the rotor-stator wheel-space of a turbine stage. Sealing air, bled from the compressor, is also used to reduce or prevent ingestion, but this can be at the expense of stage efficiency.

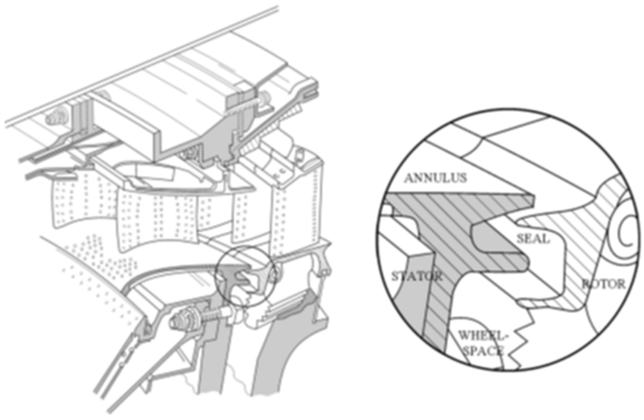
This paper summarises recent research into the computation of the fluid dynamics and heat transfer of ingestion carried out at the University of Bath. 3D Unsteady Reynolds-averaged Navier-Stokes (URANS) simulations have been carried out, and much more economical simplified steady state computational models have also been tested. The results of the computations are compared with experimental measurements (also made at the University) of pressure, tracer gas concentration based sealing effectiveness, swirl and heat transfer for different generic and also engine-representative rim seals. The computations allow insight into the flow physics of ingestion and factors affecting the most efficient use of sealing air, as well as providing information to support the development of the theoretical models of ingestion that are useful to engine designers. The experimental test facility permits measurement and ranking of sealing effectiveness for a range of different rim seal configurations.

NOMENCLATURE

b	outer radius of rotor and stator discs
C_p	non-dimensional annulus pressure coefficient
$C_{w,0}$	nondimensional sealing flow rate ($= \dot{m}/\mu b$)
G_c	seal-clearance ratio ($= s_c/b$)
$\dot{m}$	mass flow rate
Nu	local Nusselt number on rotor disc
p	static pressure
r	radius
Re_ϕ	rotational Reynolds number ($= \rho\Omega b^2/\mu$)
S	axial distance between rotor and stator
s_c	seal axial clearance
V_ϕ	tangential component of velocity
z	axial distance (from stator)
β	swirl ratio ($=V_\phi /\Omega r$)
ε	concentration-based sealing effectiveness
Φ_0	sealing flow parameter ($=C_{w,0}/2\pi\ G_c\ Re_\phi$)
Φ_{min}	minimum value of Φ_0 to seal wheel-space
λ_T	turbulent flow parameter ($=C_{w,0}\ Re_\phi^{-0.8}$)
θ	non-dimensional circumferential location across a vane pitch
μ	dynamic viscosity
ρ	density
Ω	angular speed of rotating disc

INTRODUCTION

Fig. 1 illustrates the rim seal used at the clearance between discs in the turbine stages of a gas turbine, in order to reduce the potentially damaging ingestion (or ingress) of the hot mainstream annulus gas flow into the wheel-spaces formed by the adjacent stator and rotor discs. Protection of critical regions of these rotor and stator components is further afforded by the use of sealing (or purge) air drawn from the compressor, but the efficient use of this air contributes significantly to stage efficiency as well as preserving component life. Small improvements in sealing design can therefore result in substantially better overall engine performance.



(a) (b)

Fig. 1 (a) typical high-pressure gas turbine stage; (b) detail of rim seal [4].

At the University of Bath, the flow and heat transfer in the wheel-spaces between rotor and stator discs, with and without ingestion, has been studied for the last twenty five years. Throughout these studies, computational work has been used to supplement experimental campaigns, and has provided extra detail for the development and application of theoretical models of flow and heat transfer. Experimental research carried out at the University into ingestion, which has been in progress since around 2007, has recently been summarised by Scobie *et al.* [1], and the present paper describes results from corresponding computational studies. Experimental and computational results have been presented at the annual ASME Turbo Expo congress; computational work has also been presented at the IGTC meetings held in 2007 and 2011.

The experimental facility, described in more detail by Scobie *et al.* [1], involves a (relatively small) 10 mm radius annulus, containing 32 stationary vanes and 41 rotating blades, radially outwards of a rotor-stator wheel-space having an outer radius b = 190 mm. So-called externally induced (EI) ingress and egress occurs circumferentially in to and out of the wheel-space through the rim seal between the rotor and stator discs, due (respectively) to the relatively higher and lower pressures in the annulus caused by the flow past the stator nozzle guide vanes (NGV) and the rotating blades (RB), as illustrated by computational results in Fig. 2a, compared with the pressure in the wheel-space. The rim seal in the Bath experimental rig is located mid-way between the trailing edge of the stator vane and the leading edge of the blade, see Fig. 2b; the modular design of the rig permits various rim seal configurations to be studied. Fig. 2b illustrates a simple axial clearance seal (for which s_c = 2 mm), used as the baseline reference for classification and quantification of the performance of other seal geometries.

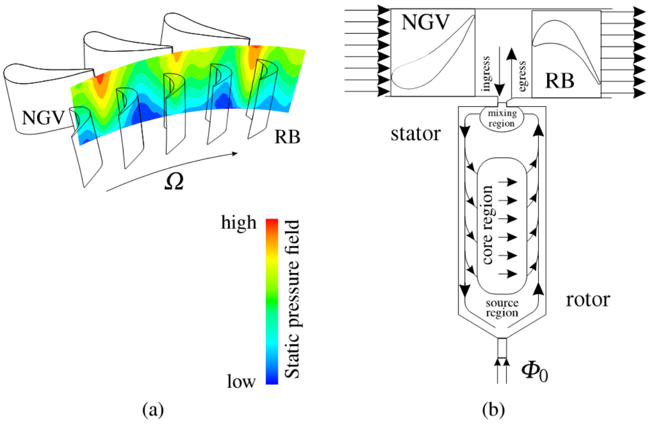


Fig. 2 Flow behaviour in (a) the annulus and (b) wheel-space [3].

In order to permit fundamental studies of the fluid mechanics of ingestion, and to avoid the need for the use of a dynamometer in the experiments, the rotating blades were of symmetric (NACA 0018) aerofoil section and were unloaded at the annulus flow experimental design condition. The rotor disc was rotated by an electric motor at speeds up to 4000 rpm, and sealing air mass flow rates were used so that values of the turbulent flow parameter, λ_T, matched those typically found in gas turbine cooling systems, so that the flow structures in the wheel-space in the rig are expected to be representative of those occurring in the wheel-spaces in engines. This rotor-stator flow structure is illustrated schematically in Fig. 2b. Sealing flow, entering the wheel-space at low radius, is drawn into the boundary layer on the rotor and is “pumped” radially outwards. Mixing occurs with ingested fluid in the region within and around the rim seal, and the mixed fluids are entrained into the boundary layer flowing radially inward on the stator. Entrainment from the stator to the rotor boundary also occurs, through the rotating core region away from the two discs, where radial velocities are very small.

The experimental rig has been used to measure radial distributions of swirl ratio, β, in the wheel-space, and also radial distributions of a concentration based sealing effectiveness, ε, on the stator disc surface. The effectiveness measurements were obtained by sampling levels of CO_2 tracer gas introduced with the sealing flow. Scobie *et al.* [1] also describe briefly how these effectiveness measurements can be related to a theoretical model of ingestion, through which the sealing effectiveness for different rim seal geometries can be shown to vary with the value of a non-dimensional sealing flow parameter, Φ_o. Measurements have also been made of the circumferential pressure distribution in the mainstream annulus; the theoretical model predicts that the amount of ingestion into the wheel-space for a fixed sealing flow rate depends upon the magnitude of the “peak-to-trough” variation of these pressures.

The different computational studies described in this paper were undertaken to address the following research questions: (i) Can 3D Unsteady Reynolds-averaged Navier-Stokes (URANS) computations capture the interactions between ingestion into the wheel-space and purge flow sufficiently accurately to be used for rim seal design and optimisation? (ii) Can simplified computational models capture the main features of ingestion sufficiently realistically to be potentially useful to engine designers? (iii) Can the effects of ingestion on rotor heat transfer be predicted accurately?

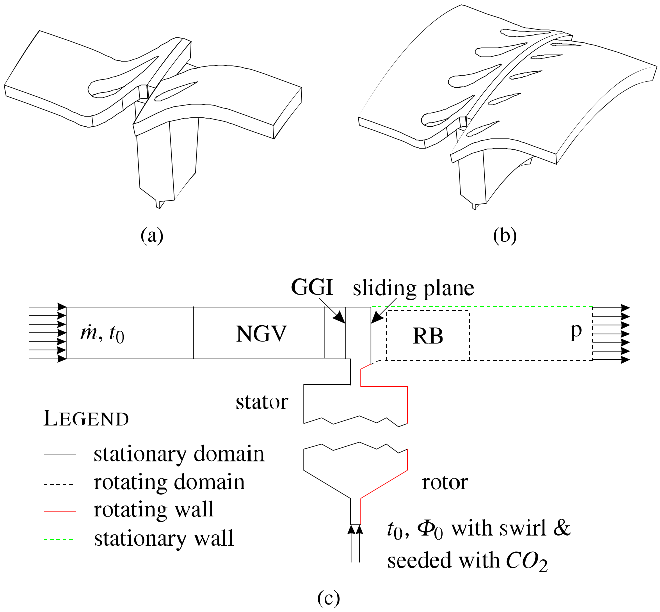


Fig. 3 Illustration of models used for URANS computations: (a) single vane, single blade model; (b) 4 vane, 5 blade model; (c) computational domain and boundary conditions [3].

All of the computations described here were carried out with the commercial code ANSYS/CFX, using its version of the SST model of turbulence and computational meshes for which the near wall mesh resolution satisfied the requirements for the use of low Reynolds number turbulence modelling treatments in the wheel-space. More specific details of the computational models are given in each relevant publication.

UNSTEADY COMPUTATIONS OF INGESTION

Zhou *et al.* [2] reviewed published computational studies of ingestion, including Large Eddy Simulation and turbulence modelling applied to both experimental rig and turbine engine operating conditions and seal geometries, and carried out the first computations based on the geometry of the Bath experimental rig. 3D Unsteady RANS computations suggested that the presence of the symmetric rotating blades augmented the circumferential time-average static pressure asymmetry in the annulus caused by the stator vanes, but that the effects were small. The performance of simplified steady computational models was also considered, and the predicted trends in the variation of sealing effectiveness with sealing flow rate (for the baseline axial clearance seal geometry) were found to be consistent with the behaviour suggested by a new theoretical model of ingestion. The study of Zhou *et al.* was undertaken in advance of the first experimental results from the rig at Bath becoming available.

Teuber *et al.* [3] subsequently carried out 3D URANS computations for selected ingestion cases that had been studied experimentally using the Bath rig. Results from a single vane, single blade model, corresponding to 32 vanes and 32 blades and illustrated in Fig. 3a, were compared with those from a more representative 45 degree sector domain model containing 4 vanes and 5 blades (Fig. 3b), from which it was concluded that the more economical single vane, single blade approximation was sufficiently realistic. Fig. 3c illustrates some further details and boundary settings for the model. Computations of the ingress-driving peak-to-trough pressure variations in the annulus showed reasonable agreement with measurements made both at the stationary vane trailing edge and on the outer casing of the annulus radially outward of the mid-point of the seal. The measured radial variation of swirl ratio was also predicted accurately by the transient computations for the generic axial clearance rim seal geometry, indicating that the fluid mechanics and the mixing process were properly represented despite the URANS computations not fully reproducing measurements of sealing effectiveness in the inner part of the wheel-space. For both the axial and a radial clearance seal geometry, good predictions of the minimum amount of sealing flow required to seal the wheel-space from ingestion were obtained, compared with those determined experimentally. For the axial clearance seal, difficulties were encountered with obtaining reliable computational results at very low sealing flow rates (and consequently high levels of ingestion) and for flow rates at the other extreme for which the wheel-space was almost fully sealed. It was suggested that this might be due to instabilities occurring in the flow structures in the region of the rim seal. Overall these computations were successful, but were also very time-consuming; although computed pressures monitored at selected locations in the annulus and the wheel-space converged to periodic behaviour within half a revolution of the rotor-disc, it was found that 8 full revolutions were required to be computed before monitored values of a passive scalar concentration variable (used to compute sealing effectiveness) reached convergence on the surface of the stator. Computations at each test condition required around two weeks of continuous computing time using eight processors on a parallel computer.

The principal objective for the computations carried out by Teuber *et al.* [3] was the extrapolation of findings from experiments carried out at comparatively low speed (and incompressible flow) rig conditions to the higher rotational speeds, annulus flow Mach numbers and ingestion-driving pressure variations that occur at the operating conditions in engines, where measurements of ingestion are usually either very limited or entirely unavailable. Theoretical modelling suggests that this extrapolation might be achieved by scaling the minimum non-dimensional sealing flow rate required to prevent ingestion, known as Φ_{min}, proportionally according to the different peak-to-trough annulus pressure variations in the experimental rig and engine situations. The extrapolation method devised and described by Teuber *et al.* [3] showed good agreement with computed values of Φ_{min} over the annulus flow Mach number range investigated. It was proposed that this method could be used by the designers of secondary air systems to extrapolate the measured sealing effectiveness for a particular rim seal, obtained from an experimental test facility, to a geometrically-similar engine. The advantage of the method is that good estimates of annulus pressure distributions can be obtained relatively rapidly by transient or possibly even steady-state computations; the highly time-consuming simulation of ingress at the compressible flow engine conditions is not required. (A caveat is that the method was only tested for subsonic annulus flows; the onset of shockwaves in the transonic regime might have a significant influence.)

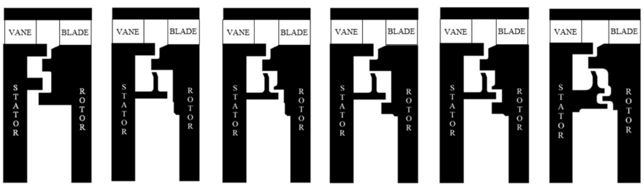


Fig. 4 Evolution of an improved rim seal design [4].

The study of Teuber *et al.* was carried out as a joint project with an industrial partner, with computations being carried out at the company. The project extended further to the design and computational evaluation of an improved rim seal design for use in engines. The final results of this study gave rise to a patent application for the design, and the work has been described by Scobie *et al.* [4]. Fig. 4 shows the evolution of the seal design, resulting from a sequence of separate computations that began with a generic double radial seal geometry, Fig. 4a, as tested previously experimentally at Bath, and ending with a refined "angel-wing" design, Fig. 4f, incorporating a modified buffer volume in the outer part of the wheel-space. Computations were based on an existing engine geometry and were carried out at engine operating conditions; each 20-node parallel computation required around 5 weeks to complete. Fig. 5 and Fig. 6 illustrate respectively the computed secondary flow structure and predicted performance of the final rim seal design at the point in the unsteady cycle corresponding to peak ingestion through the seal. The buffer volume acts to attenuate the variation in circumferential pressure which is the driver to force ingress further through the intermediate seal. The computed distribution of concentration-based sealing effectiveness in Fig. 6 (at the point of maximum ingestion) would correspond to regions of higher and lower temperature under engine conditions, with low values of effectiveness indicating large amounts of ingested fluid (which would be at high temperature in the engine). Fig. 5 shows that the mixed-out ingress in the intermediate seal is opposed by the sealing flow pumped radially outwards by the rotating disc. The buffer volume contains the majority of the ingested fluid, while a recirculating counter-rotating vortex, encouraged by the shape of the angel wing (and supplemented by a small leakage flow), prevents ingested fluid from reaching the stator surface. The rotor boundary layer flow also delivers coolant to the underside of the blade platform.

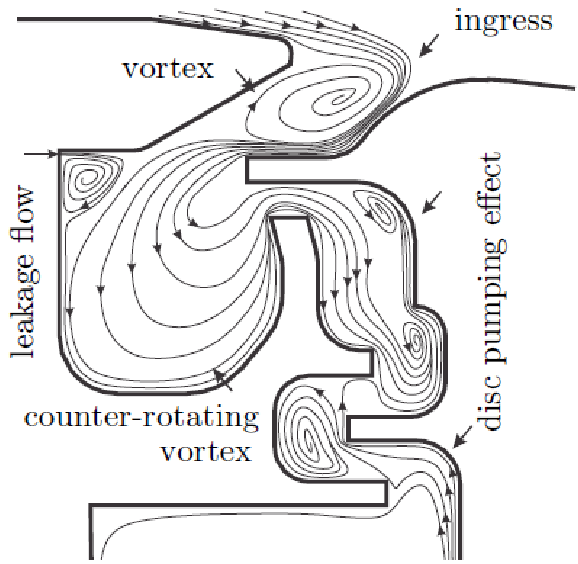


Fig. 5 Case of maximum ingress for improved seal design; computed secondary flow stream lines [4].

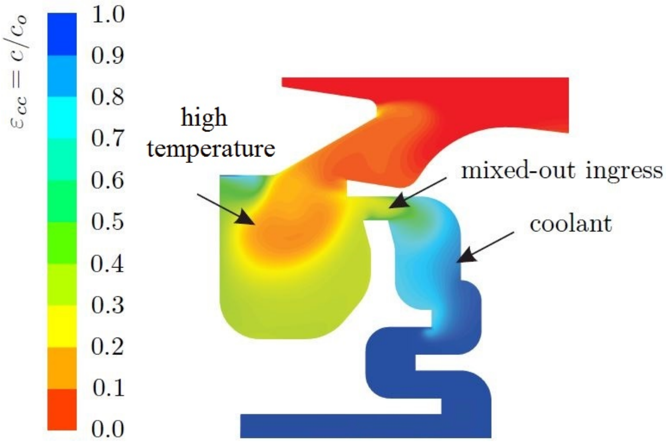


Fig. 6 Case of maximum ingress for improved seal design; contours of computed concentration-based sealing effectiveness ε [4].

The improved performance of this final rim seal design over earlier variants was confirmed by experiments carried out in the Bath rig, demonstrating very successfully this method of combining computation and detailed experiments to translate measurements made under benign rig conditions to the engine situation.

3D STEADY COMPUTATIONS

Recognising the very long computing times required by unsteady ingestion computations, Lalwani *et al.* [5] simplified the problem configuration by carrying out computations for a model of the Bath rig involving one stator vane pitch but *without* rotating blades. (Zhou *et al.* [2] had previously carried out similar computations and shown that the time-varying effect of the (symmetric) rotating blades on the ingestion-driving pressure distribution in the annulus was small). An example of the agreement between computed and measured pressure distributions in the annulus is shown in Fig. 7, in the form of a non-dimensional pressure coefficient C_p. The comparison is made at the location A in the annulus illustrated in the inset figure, this is on the stator platform at the trailing edge of the stator vanes. Computational results are shown for two generic radial seal geometries. Teuber *et al.* [3] obtained similar agreement for both the time averaged distribution from unsteady computations and from a "frozen rotor" steady computation (used as the starting condition for the unsteady case) where the rotor blades were in a fixed circumferential location relative to the vanes. (Less good agreement was obtained by Lalwani *et al.* at the other location where measurements were made, on the outer radius of the annulus outward of the centre of the seal clearance and therefore closer to the location of the rotating blades in the experimental rig.)

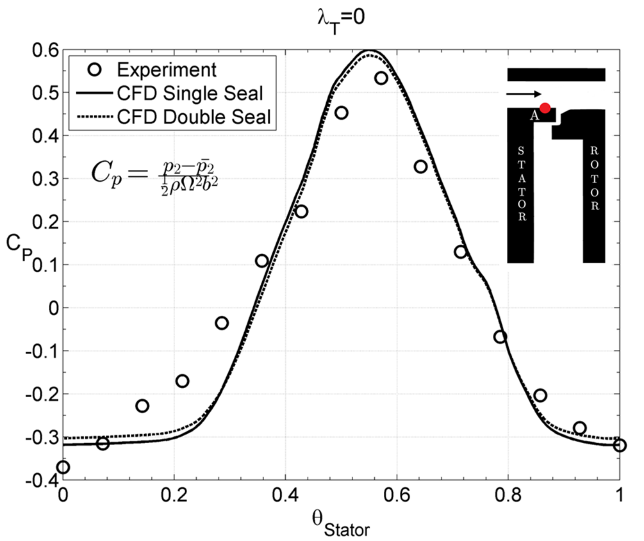


Fig. 7 Comparison between measured and computed circumferential distribution of static pressure in the annulus at location A (hub) at Re_ϕ=8.2×10^5 [5].

By modelling the wheel-space in a frame of reference rotating with the rotor disc, Lalwani *et al.* found that their 3D steady computations gave reasonably good predictions of ingestion for a range of cases from the Bath rig experimental programme. Fig. 8 shows (with lines and symbols respectively) the computed and measured variation of swirl ratio with non-dimensional radius in the wheel-space for a generic radial clearance seal and for a range of values of sealing flow rate, at a rotational Reynolds number Re_ϕ=8.2×10^5. The radial seal geometry is illustrated in the contour plot to the right of the figure showing the distribution of swirl ratio, β, in the wheel-space for the particular case of no sealing flow (λ_T = 0); the square red symbols superimposed on this figure show the locations (at z/S = 0.25) where the total pressure measurements were made that were used to determine the tangential velocity V_ϕ.

The computed swirl distributions show very good qualitative and reasonably good quantitative agreement with the measurements, suggesting good prediction of the amount of ingestion and of the mixing inside the wheel-space using the steady-state CFD model. Radially outward of non-dimensional radius r/b = 0.95 (where experimental information is not available) Fig. 8 shows that the computed swirl ratio increases rapidly due to the high swirl associated with the fluid ingested from the annulus (where β = 1.8 approx. for the mainstream flow downstream of the vanes) entering the wheel-space. Fig. 8 also shows that, as the flow rate of the (non-swirling) sealing air is increased, there is a reduction in the swirl ratio of the "rotating core" of fluid outside the boundary layers on the rotor and stator discs. Increasing the sealing flow rate pressurises and progressively seals the wheel-space from ingestion, and the swirl ratio near the periphery of the wheel-space reduces as the amount of ingress from the annulus reduces. The parametric variation of swirl ratio with λ_T is represented well by the computations.

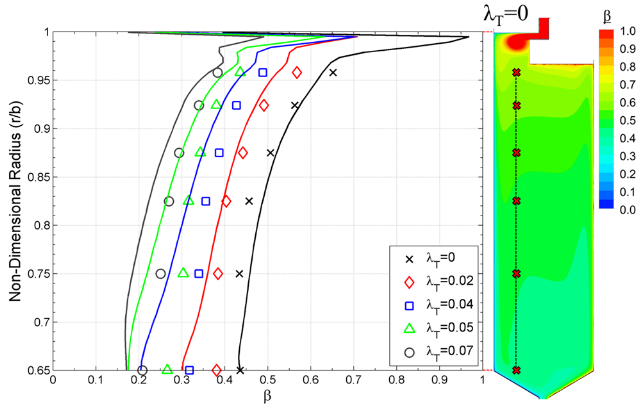


Fig. 8 Variation of swirl ratio with radius for single radial seal for different λ_T at Re_ϕ=8.2×10^5 [5].

Figure 9 shows the computed and measured radial variation of sealing effectiveness at the stator surface for three different sealing flow rates, also at Re_ϕ=8.2×10^5. The computed results show good agreement with the measured distributions for r/b < 0.9. The experiments show that, for each value of λ_T, the effectiveness is approximately constant with radius. This suggests that almost complete mixing occurs, between the ingested fluid and the flow inside the wheel-space, in a region very close to the rim seal. At high radii, (0.9 < r/b < 1) the computations differ from the measurements in both magnitude and behaviour, indicating that the combination of turbulence modelling and steady state simplifications may not capture fully the mixing processes within the seal. The contour plot to the right of Fig. 6 shows the computed distribution of sealing effectiveness. This illustrates the extent of the mixing region in the computations.

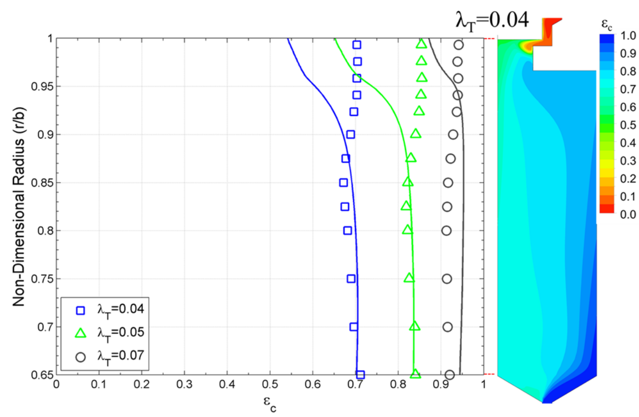


Fig. 9 Variation of sealing effectiveness with radius for single radial seal for different λ_T at Re_ϕ=8.2×10^5 [5].

Figure 10 shows three-dimensional features of the computed swirl ratio in the radial-tangential plane at z/S=0.5 for a generic double-radial clearance seal (this is the seal used as the starting point for the evolution of a new seal design as illustrated in Fig. 4). Non-axisymmetric behaviour is confined to the outermost regions of the seal, further illustrating that rapid mixing occurs between the ingested fluid and the flow in the wheel-space. The uniformity of effectiveness at lower radii indicates that full mixing has occurred and reflects the results shown in Fig. 8 for a single radial seal, though the sealing performance is improved in the inner part of the wheel-space using the double seal. The non-uniformity of the swirl ratio contours in the annulus also indicates that significant interaction can occur between the annulus flow and the lower swirl egress flow leaving the wheel-space mixing region through the rim seal.

These results are sufficiently similar to the time-averaged results obtained at greatly increased computational cost in other studies to suggest that the steady-state modelling of a 3D turbine stage used here may be a realistic tool for the engine designer, to provide quantitative prediction of flow structures and qualitative insight into the effects of ingestion on the flow within the wheel-space when there is a lack of experimental data.

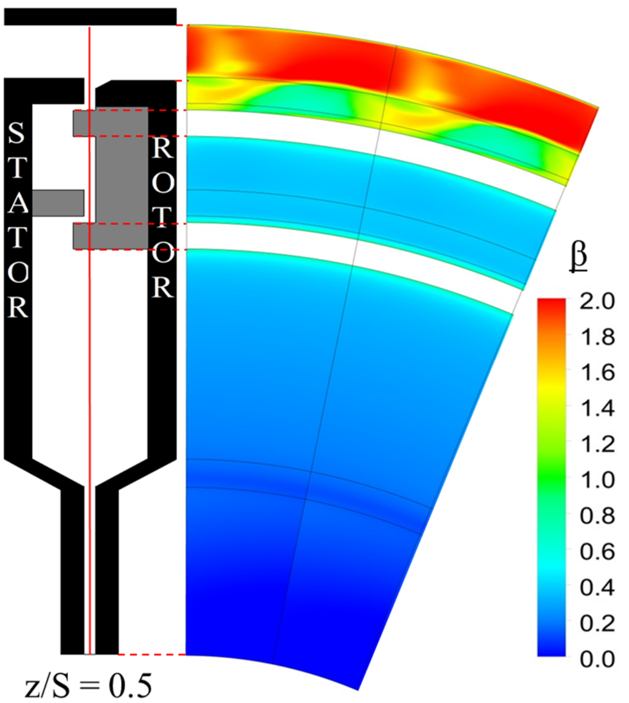


Fig. 10 Computed swirl ratio at z/S=0.5 for double radial clearance seal for λ_T =0.04 at Re_ϕ=8.2×10^5 [5].

AXISYMMETRIC COMPUTATIONS OF FLOW AND HEAT TRANSFER

Wang and Wilson [6] studied the effects of ingestion on flow and heat transfer in the wheel-space of the Bath experimental rig by *prescribing* rather than directly computing ingress at the rim seal clearance, by using a modified seal clearance boundary in a simplified axisymmetric model of the wheel-space only. This avoided the sensitive coupling between the wheel-space domain and the annulus (for which the computational requirements are very different). Wang and Wilson used measurements of sealing effectiveness (and features of the theoretical model of ingestion as mentioned in the introduction) to estimate the mass flow rate, associated swirl and inlet values for other variables for the "ingested" fluid entering the wheel-space, using an empirical mass-weighted averaging technique [6]. Using this method, measured values of swirl ratio in the wheel-space could be computed with reasonable accuracy, and the qualitative trends for the sealing effectiveness measured on the stator could be reproduced for generic rim seal geometries. Fig. 11 shows the boundary condition modifications (for an axial clearance seal) and illustrates the computed flow structure and distribution of sealing effectiveness in the wheel-space. The results indicate that the effects of ingestion on the rotor-stator behaviour in the wheel-space are small, even for (as in the case shown) relatively low sealing flow rates and therefore significant amounts of ingestion. Fig. 11 shows that, as expected, the sealing flow is drawn onto the rotor (on the right in the figure) at low radius, while the "ingested" fluid entering through the prescribed ingestion inlet boundary at the seal clearance location at the periphery of the system is entrained into the boundary layer flowing radially inwards on the stator. This gives rise to "buffering" of the rotor-disc from the ingested fluid (which in the engine would be hot and potentially damaging) by the rotor boundary layer augmented by the sealing flow. (The outlet boundary illustrated at the outer radius on the rotor side in Fig. 11 allows the sealing and "egress" flow to leave the system in the axisymmetric model.)

Wang and Wilson [6] found that the effects of wheel-space geometry on the computed flow and heat transfer were also small, with computations based on the Bath ingestion rig geometry giving very similar results to those for a different geometry (for which b = 381 mm rather than 190 mm) used in earlier rotor-stator experiments without ingestion, for which validation measurements for both flow and rotor disc heat transfer were available in the literature. The computations afforded good predictions of these measurements.

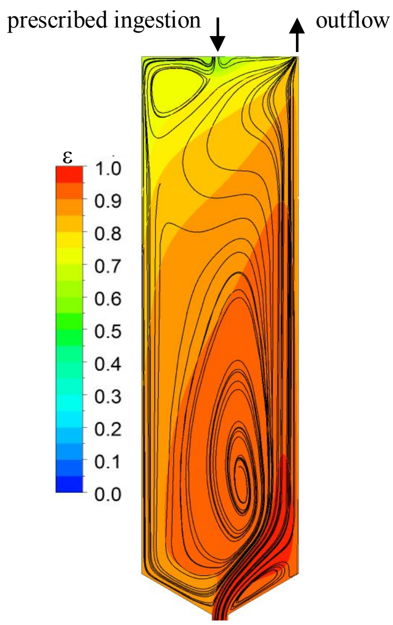


Fig. 11 Computed flow structure and distribution of sealing effectiveness for the prescribed ingestion model for λ_T =0.081 [6].

Wang and Wilson [7] developed the prescribed ingestion technique further in order to make comparisons between computed rates of heat transfer and measurements (made by heating the sealing air and using a transient thermochromic liquid crystal technique) at the surface of the Bath ingestion rig rotor disc, for two values of sealing flow rate. Computations of these measurements had not previously been reported. Fig. 12 shows that the computations are able to reproduce features of the measured distribution of heat transfer on the rotor although the levels of local Nusselt numbers are over-predicted. Fig. 12 also shows a significant effect of the thermal boundary conditions assumed for the stator on the computed rotor disc heat transfer. (Subsequent (unpublished) studies found that the effects of the estimated inlet swirl and temperature used in the prescribed ingestion boundary conditions are very small.) The measured Nusselt numbers shown in Fig. 12 are higher for the higher λ_T case, i.e. for the higher (non-swirling) sealing flow rate; this is consistent with the corresponding reduction in the swirl ratio in the rotating core of fluid in the wheel-space. Both sets of Nusselt number measurements are significantly lower than those involved in the validation case considered by Wang and Wilson [6] for a rotor-stator system with radial outflow in the absence of ingestion; this is consistent with the increase in swirl ratio in the system due to the higher swirl associated with the ingested fluid.

Wang [8] showed that the prescribed ingestion technique could be applied successfully to other seal geometries such as the single and double radial seals also studied by Lalwani *et al.* [5]. Very economical computations of the wheel-space such as these could be of use to engine designers, who require estimates of the magnitude and distribution of swirl and heat transfer in order to develop correlations for use in design codes.

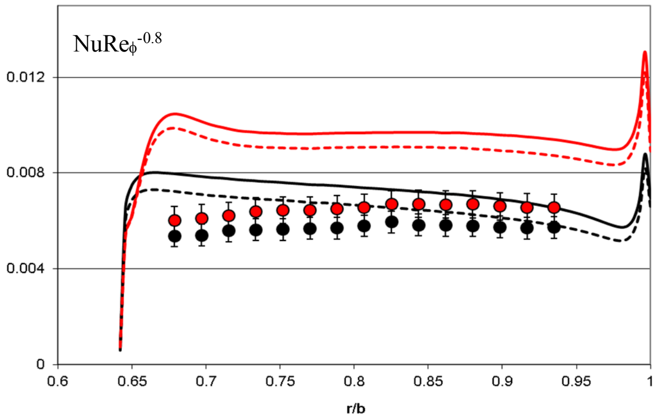


Fig. 12 Computed radial variation of Nusselt number for $Re_{\phi}=8.2\times10^5$ compared with experiments (solid lines – adiabatic stator, dashed lines – fixed temperature stator) [7].

FUTURE WORK

All of the studies described above were based upon the geometry and measurements from the single stage rig at the University of Bath. Experiments are now underway on a new 1.5 stage (vane-blade-vane) rig having similar characteristics to the existing rig but giving the opportunity to study the so-called downstream wheel-space, for which the relative orientation of the rotor and stator is reversed (compared with that shown in Fig. 1) and where the driving annulus pressure variations for ingestion are expected to be due to the flow over (loaded) rotating blades rather than stationary vanes.

In addition to the unsteady 3D computations described above, Teuber [9] also carried out a computational study of rotor blade endwall contouring (EWC) with the aim to better manage the interaction between the egress flow leaving the wheel-space and the annulus mainstream flow, potentially improving stage efficiency. This work will now be extended as part of a new programme of work funded jointly by industry and the UK Engineering and Physical Sciences Research Council. The same highly effective method of working demonstrated in some of the studies described here, with experiments being carried out using an experimental rig and computations being applied to the engine situation, will be used once again.

CONCLUSIONS

This paper has summarised published computational studies of ingestion into the wheel-space of a gas turbine stage based on the details of an experimental research rig at the University of Bath. Computed variations of pressure, swirl velocity, concentration-based sealing effectiveness (as a measure of ingestion) and heat transfer have been compared with measurements made using the rig. 3D Unsteady Reynolds-Averaged Navier-Stokes computations have allowed experimental findings from the comparatively benign conditions of the low speed, incompressible flow experiments to be translated with some confidence to engine operating conditions, though at the expense of very long computing times due to the different requirements for the computation of the flow in the annulus and in the wheel-space.

Steady flow computations carried out for simplified versions of the Bath rig geometry show qualitative and quantitative agreement with measurements at greatly reduced computational cost. Such computations may be useful to engine designers in understanding the effects of ingestion into the wheel-space for particular rim seal designs, and in offering an economical method for providing information for the development of flow and heat transfer correlations as used in industrial design tools. New computational and experimental work is now underway to further exploit the techniques and methods already demonstrated, for application of the results of fundamental research into ingestion to the design and improvement of sealing and cooling systems in engines.

ACKNOWLEDGEMENTS

The URANS computations described here were carried out using facilities at the industrial partner as part of a UK Technology Strategy Board Knowledge Transfer Partnership. Other computations were carried out using the Aquila High Performance Computing Facility at the University of Bath.

REFERENCES

[1] Scobie, J. A., Sangan, C. M., Owen, J. M. and Lock, G. D., 2015, "An Overview of Hot Gas Ingestion Research at the University of Bath," *Proceedings of 11th European Turbomachinery Conference (ETC 2015)*, March 23-27, Madrid.

[2] Zhou, K., Wilson, M., Owen, J. M. and Lock, G. D., 2013, "Computation of Ingestion through Gas Turbine Rim Seals," *Proceedings of the Institution of Mechanical Engineers Part G: Journal of Aerospace Engineering*, 227 (7), pp. 1101-1113

[3] Teuber, R., Wilson, M., Lock, G. D., Owen, J. M., Li, S., and Maltson, J. D., 2013, "Computational Extrapolation of Turbine Sealing Effectiveness from Test Rig to Engine Conditions," *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 227 (2), pp. 167-178.

[4] Scobie, J. A., Teuber, R., Li, Sangan, C. M., Wilson, M. and Lock, G. D., 2015, "Design of an Improved Turbine Rim-Seal," *ASME paper GT2015-42327*, to appear in *ASME J. Engineering for Gas Turbines and Power*

[5] Lalwani, Y., Sangan, C. M., Wilson, M., Owen, J. M. and Lock, G. D., 2015, "Steady Computations of Ingress through Gas Turbine Rim Seals," *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 229 (1), p. 2-15

[6] Wang, L. and Wilson, M., 2012,"Computations of flow and heat transfer in a rotor-stator system with externally-induced ingestion," *International Journal of Gas Turbine, Propulsion and Power Systems*, 4 (1), pp. 10-18,

[7] Wang, L. and Wilson, M., 2014, "Flow and Heat Transfer in Rotor-Stator Systems with and without Ingestion," *Proceedings of 15th International Heat Transfer Conference*, August 10-15, 2014, Kyoto, IHTC15-8457.

[8] Wang, L., 2014, "Study of Gas Turbine Ingress Using Computational Fluid Dynamics," *PhD Thesis*, University of Bath.

[9] Teuber, R., 2014, "Computation of Rim-Sealed Ingestion for Gas Turbines," *PhD Thesis*, University of Bath.

APPENDIX: SELECTED COMPUTATIONAL STUDIES BY OTHER WORKERS

Computations of ingestion have been carried out by researchers from many other organisations, based on different experimental rigs or engine geometries and using different computational methods to those that have been tested at the University of Bath. The most ambitious of these studies to date include those by Wang *et al.* [A1] and Mirzamoghadam *et al.* [A2], who each carried out full 360 degree (rather than circumferentially-periodic sector) computations for a turbine stage test rig, and O'Mahoney *et al.* [A3] who were the first to describe results from Large Eddy Simulation (LES) investigations of ingestion. Other work includes that of Mirzamoghadam *et al.* [A4], who used the NUMECA International Fine/Turbo code and both mixing-plane and non-linear harmonic balance methods to reduce computing time for URANS calculations of an engine turbine stage geometry at test conditions ($Re_\phi=4.42\times10^6$). The sector model involved one stator vane pitch, and the Spalart-Allmaras turbulence model was used. Computational results were compared with temperature-based sealing effectiveness predictions from a proprietary 1-D network model matched to test data. The influence of rim seal overlap features on the effectiveness of the design were quantified for three different geometry configurations.

Wang *et al.* [A1] used a full 360 degree computational model based on the geometry of a rim seal experimental facility at Arizona State University (ASU). The commercial FLUENT solver was used and the SST turbulence model was selected. The turbine stage involved 22 stationary vanes and 28 rotating blades. The rotor speed was 2400 rpm giving a rotational Reynolds number of $Re_\phi=5.86\times10^5$. 1848 time steps were used for computation of one full rotation of the rotor disc, and computations at each of the three purge flow rates considered required around 6 weeks using 44 processors on a parallel computing cluster. Six revolutions of the disc were computed. These computations were carried out for an axially-overlapping radial clearance seal. Mirzamoghadam *et al.* [A2] carried out 360 degree computations for the same ASU rig geometry and rotor speed but for a different (double radial clearance) seal geometry. The air was treated as a compressible ideal gas and the SST turbulence model used included low Reynolds number correction and compressibility effects. The computational results showed that the flow remained unstable and still changing after 16 revolutions of the disc, although circumferentially averaged values of sealing effectiveness at the stator surface became constant after 11 revolutions. Details of the representation of rotation in the model was found to affect the amount of ingestion predicted, and consequently also the agreement between the computations and measurements of ingestion. Ingestion into the outer part of the rotor-stator wheel-space was over-predicted, and the comparison with measurement was poorest at the lowest sealing flow rate. As had also been found by Wang *et al.*, the number and location of the non-periodic regions of ingress and egress computed circumferentially around the rim seal changed with each revolution of the disc.

O'Mahoney *et al.* [A3] compared results from LES and URANS computations of ingestion for a rotor-stator wheel-space with an oblique chute rim seal, as commonly used in industrial turbomachinery. The solver used was based on the Rolls-Royce HYDRA CFD code. The computational domain was a 13.3 degree (one vane, two blade) circumferentially periodic sector, approximating an experimental rig that had 26 stationary vanes and 59 rotating blades. The rotational Reynolds number $Re_\phi=2.2\times10^6$ was chosen in order to make comparisons with existing experimental results. The LES calculations required 150 blade passing intervals before reliable statistics could be taken and computing times were around 20 days using 256 processors. The corresponding URANS computations required 14 days using 32 processors. The results, and comparison between modelling approaches as well as with experimental measurements suggested that, in spite of non-periodic behaviour as observed in the 360 degree URANS simulations described above, simplification of the calculation domain to a cyclically-symmetric sector is less influential than turbulence modelling for the accurate simulation of ingestion.

Ding *et al.* [A5] reported computational results for geometry and test conditions (7600 rpm, $Re_\phi=6.0\times10^6$) that matched experiments carried out using a 1.5 stage ingestion test rig at General Electric. The simplified computational modelling described included k-ε turbulence modelling and a wall function approach, with computation times of around 100 hours on 96 CPUs of an HPC cluster. The correct trends for ingestion (compared with concentration-based sealing effectiveness measurements) were predicted for the four purge flow rates considered, and time-averaged computational results were used to illustrate flow paths through the angel wing rim seal and in the trench (or buffer) cavity in the forward wheel-space of the stage. Use of the computational model was anticipated for future work on wheel-space and rim seal conceptual design.

REFERENCES FOR APPENDIX

[A1] Wang, C. Z., Mathiyalagan, S. P., Johnson, B. V., Glahn, J. A. and Cloud, D. F., 2012, "Rim Seal Ingestion in a Turbine Stage from 360-Degree Time-Dependent Numerical Simulations"; *ASME J. Turbomach*., 136(3), 031007-1 to 12

[A2] Mirzamoghadam, A. V., Kanjiyani, S., Riahi, A., Vishnumolakala, R., and Gundeti, L., 2015, "Unsteady 360 Computational Fluid Dynamics Validation of a Turbine Stage Mainstream/Disk Cavity Interaction," *ASME J. Turbomach*., 137(1), 011008-1 to 9.

[A3] O'Mahoney, T.S.D., Hills, N.J., Chew, J.W. and Scanlon, T., 2010, "Large-Eddy Simulation of Rim Seal Ingestion"; *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Scienc*e, 225 (12), pp. 2881-2891

[A4] Mirzamoghadam, A. V., Heitland, G., and Molla-Hosseini, K., 2009, "The Effect of Annulus Performance Parameters on Rotor–Stator Cavity Sealing Flow," *ASME Paper GT2009-59380*.

[A5] Ding, Z., Palafox, P., Moore, K., Chupp, R., Kirtley, K.,2013 "A New 1.5-stage Wheelspace Hot Gas Ingestion Rig (HGIR) – Part II: CFD Modeling and Validation," *ASME Paper GT2013-96021*.